RECENT PROGRESSES IN FLUID DYNAMICS RESEARCH
Proceedings of the Sixth International Conference on Fluid Mechanics
AIP Conf. Proc. Vol 1376, 467-469 (2011)

Unsteady Flow Dynamics and Acoustics of Two-Outlet Centrifugal Fan Design

I. Y. W. Wong[1,*], R. C. K. Leung[1], A. K. Y. Law[2]

[1]*Department of Mechanical Engineering, The Hong Kong Polytechnic University, Hung Hom, Kowloon, Hong Kong, China*
[2]*Raymond Industrial Limited, 18/F, Grandtech Centre, 8 On Ping Street, New Territoties, Hong Kong, China*

Email: mmrleung@polyu.edu.hk

Abstract In this study, a centrifugal fan design with two flow outlets is investigated. This design aims to provide high mass flow rate but low noise performance. Two dimensional unsteady flow simulation with CFD code (FLUENT 6.3) is carried out to analyze the fan flow dynamics and its acoustics. The calculations were done using the unsteady Reynolds averaged Navier Stokes (URANS) approach in which effects of turbulence were accounted for using k-ε model. This work aims to provide an insight how the dominant noise source mechanisms vary with a key fan geometrical paramters, namely, the ratio between cutoff distance and the radius of curvature of the fan housing. Four new fan designs were calculated. Simulation results show that the unsteady flow-induced forces on the fan blades are found to be the main noise sources. The blade force coefficients are then used to build the dipole source terms in Ffowcs Williams and Hawkings (FW-H) Equation for estimating their noise effects. It is found that one design is able to deliver a mass flow 34% more, but with sound pressure level (SPL) 10 dB lower, than the existing design .

Key words: centrifugal fan, CFD, aeroacoustics, noise reduction, unsteady flow dynamics

INTRODUCTION

It is common that many domestic products containing centrifugal fans are very noisy. The flow through centrifugal fan is highly three dimensional and unsteady; thus fan design with only trial and error basis is no longer possible. It is necessary to uncover the source of noise from this centrifugal fan in a fast and low cost manner. Computational fluid dynamics (CFD) is therefore getting popular in analyzing the fan flow dynamics. Many researchers have attempted to investigate the noise of centrifugal fan, they commonly found that the level of noise is associated with the degree of unsteadiness of the flow between fan blades and fan volute [1-6]. Design modifications were proposed by some of these authors with an aim to reduce the fan noise level. However, usually the proposed low-noise designs are contradictary to those favoring high mass flow rate requirements. In this paper, the flow dynamics through a unconventional centrifugal fan design, which contains two outlets, is analyzed using a commercial CFD code, FLUENT. Based on the CFD results, the fan noise level is predicted using FW–H equation. The objective of the works is to study the effects of fan geometrical parameters on the fan flow unsteadiness and noise generation with an aim to find out an optimal design which gives higher mass flow rate but lower noise than the existing design.

FORMULATION OF NUMERICAL PROBLEM

A schematic sketch of the fan design is illustrated in Figure 1. The key fan parameters are given in Table 1 below.

The two-dimensional simulations of the unsteady centrifugal fan flow were solved using finite volume method in URANS approach on an unstructured mesh generated within the computational domain illustered in Figure 1. The second-order, implicit segregated solver available in FLUENT with SIMPLE algorithm for pressure-velocity coupling was used. Standard k-ε model with 5% turbulence intensity was adopted for calculating the turbulence effects. Pressure inlet and pressure outlet boundary conditions with standard atmospheric pressures were set at fan inlet and outlet. No-slip boundary conditions were used on all solid surfaces. The time step size of the calculations was set to 2×10^{-4}

s. which gave $CFL < 1$ with respect to the smallest mesh of the calculations. The time-stationary CFD results were then used in the estimation of the noise source strength via FW–H equations. Calculations of existing and four new designs were carried out. The new designs are specified by the fan housing curvature C and the ratio between fan cutoff distance m separating fan housing volute tongue and impeller, and blade spacing n (Figure 2). Their combinations are listed in Table 2 below. The mass flow rate of existing design was calculated using the numerical setup mentioned and compared with experimental values. The relative error was only 1.9%.

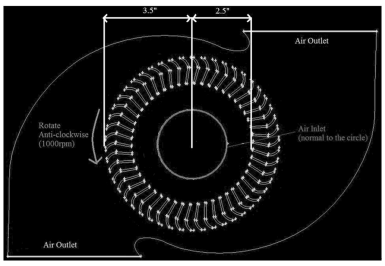

Figure 1. A schematic sketch of fan design

Table 1.

Impeller	Blade number	50 (forward curved)
	Blade thickness	1.6 mm
	Blade chord length	38 mm
	Inlet blade angle	30°
	Outlet blade angle	75°
	Impeller inlet diameter	170 mm
	Impeller outlet diameter	245 mm
	Rotational speed	1 000 rpm
Fan Housing	Inlet diameter	150 mm
	outlet size	110×65 mm
	Volute shape	Logarithmic profile

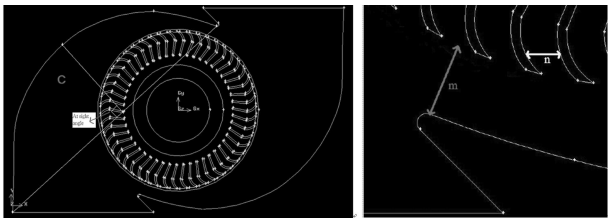

Figure 2. New designs of fan

Table 2.

Design	$m : n$	C (m)
Existing	1 : 0.8	0.099 61
1	1 : 2	0.113 4
2	1 : 0.5	0.113 2
3	1 : 2	0.090 15
4	1 : 0.5	0.090 77

RESULTS AND DISCUSSTIONS

The time-stationary static pressure distributions for existing and Design 2 are illustrated in Figure 3. It can be seen that there exist high static pressure regions in the tip regions of the volute tongues. The pressure magnitude appears to be insensitive to the design change so it effectively creates blocking to the flow. Similar patterns can also be observed in other new designs. Figure 4 shows the time-stationary vorticity distribution for Design 2. High vorticity regions are found around the blades and volute tongues which show that the noise sources in these regions are stronger according to FW–H theory.

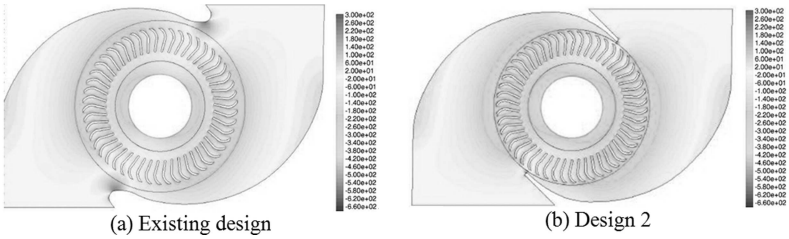

Figure 3: Static pressure distribution

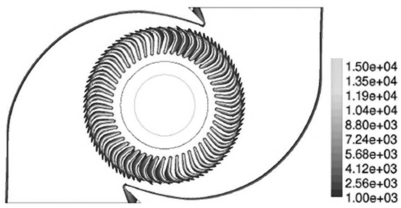

Figure 4: Vorticity distribution

The calculated mass flow rates of all designs are illustrated in Figure 5. Here $R = m/n$ with n fixed. Obviously Design 1 attained maximum increase in flow rate (∼34% more). This may be due to the fact that a larger cutoff distance reduces the recirculation of the flow which essentially reduces the flow blocking in the fan. From Figure 6(a), the recirculation zone is clearly observed in the volute tongue region. The zone also extends and interact with the rotating impeller blades so the flow blocking is strong there. In Design 1, the tongue geometry is so modified that the flow is more streamlined and easier to leave the outlet (Figure 6(b)). The flow blocking more or less disappears and results in a higher average outlet velocity and a higher mass flow rate.

According to FW–H equation [6], the acoustic source strength is dependent on time variations of fluid forces induced on the rotating impeller blades. The propagation of the generated acoustic wave is determined by the boundary geometry and acoustic conditions imposed by the housing. In the present study, only the variation of SPL with fan design change is concerned. Therefore, the change in SPL is estimated from the change of overall force coefficients $C_{\text{total}}(t) = \sqrt{C_{\text{L}}^2(t) + C_{\text{D}}^2(t)}$, where C_{L} and C_{D} are the lift and drag coefficients respectively, in the expression of source strength.

Figure 7 shows a comparison of the spectra of new and existing designs. It can be observed that the in existing dominant C_{total} fluctuation occurs at 33.33 Hz which is equal to double of impeller rotation frequency. The double is simply due to the presence of two volute tongues. In the new designs, the dominant fluctuation however shifts to follow the impeller rotation frequency, i.e. 16.67 Hz whereas other peaks are almost at the same levels as in existing design. Figure 8 shows the difference in *SPL* obtained in all new designs. It can be seen that maximum reduction of 2.22 dB is obtained from Design 1. Other designs give zero or an increase in *SPL*. Based on this result, a fan prototype of Design 1 was fabricated and tested in anechoic chamber. It was found that a 10 dB was recorded at 1 m away from the fan which provides a piece of good evidence of the correctness of numerical prediction.

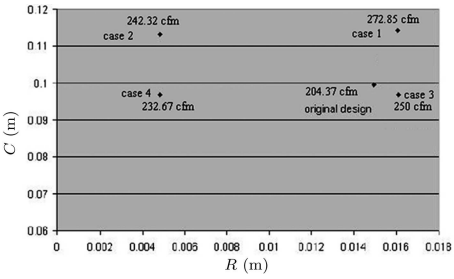

Figure 5: Variations of mass flow rate with *C* and *R*

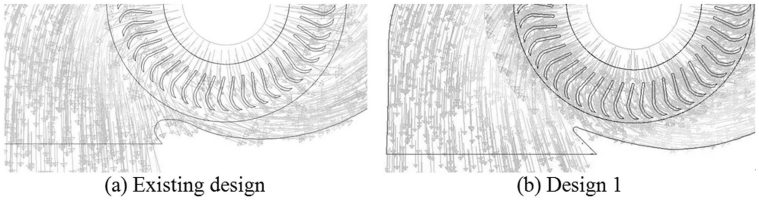

Figure 6: Velocity distribution at outlet

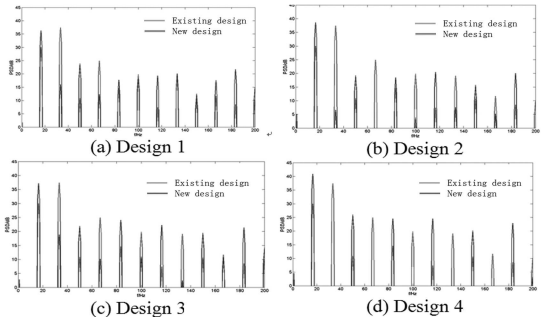

Figure 7: Spectra of overall force cofficients. Existing design: red line; new design: blue line

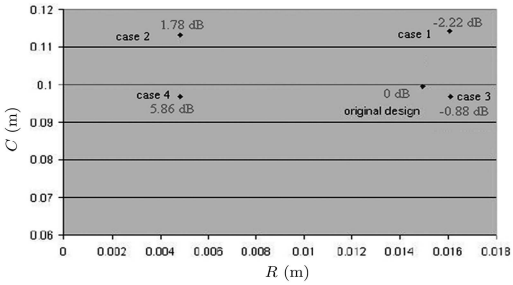

Figure 8: Variations of *SPL* difference rate with *C* and *R*

CONCLUSIONS

In this paper, a numerical study of flow dynamics and acoustics of an industrial centrifugal fan with two outlets has been reported. The flow dynamics was calculated using URANS on finite volume method available from commercial CFD code and its acoustic generation was predicted using the source terms in FW–H equation. Analyses with existing and four new fan designs, specified with fan housing curvature and the cutoff distance at volute tongue, were carried out. Finally, Design 1 was found able to increase the mass flow rate and reduce *SPL*. The result was also confirmed with experiments.

Acknowledgements

The support given by the University Industry Collaboration Programme, Innovation and Technology Fund of the Government of Hong Kong SAR Government (UIT/106) is gratefully acknowledged.

REFERENCES

1. Younsi M, Bakir F. 3D unsteady flow in a centrifugal fan: impeller-volute interaction. *J. Comput. Appl. Mech.*, 2007; **8**: 211-223
2. Jeon WH. A numerical study on the effects of design parameters on the performance and noise of a centrifugal fan. *J. Sound Vib.*, 2003; **265**: 221-230
3. Tajadura RB, Suarez SV, Cruz J PH. Noise prediction of a centrifugal fan: numerical results and experimental validation. *J. Fluid Engg.*, 2008; **130**: 091102
4. Liu Q, Qi D, Tang H. Numerical calculation of centrifugal fan noise. *J. Mechanical Engg. Sci.*, 2007; **221**: 1321-1332
5. Suarez SV, Tajadura RB, Morros CS, Garcia BP. Reduction of the aerodynamic tonal noise of a forward-curved centrifugal fan by modification of the volute tongue geometry, *Appl. Acoust.*, 2008; **69**: 225-232
6. Ffowcs Williams JE, Hawkings DL. Sound generation by turbulence and surfaces in arbitrary motion. *Phil. Trans. Roy. Soc. A*, 1969: **264**: 321-342